REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*
April 2012

2. REPORT TYPE
Viewgraph

3. DATES COVERED *(From - To)*
April 2012- June 2012

4. TITLE AND SUBTITLE
Numerical Simulation of Confined Multiple Transverse Jets

5a. CONTRACT NUMBER
In-House

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)
F. Davoudzadeh, D. Forliti, A. Lee, H. Vu

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER
33SP0795

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Air Force Research Laboratory (AFMC)
AFRL/RQRC
10 E. Saturn Blvd.
Edwards AFB CA 93524-7680

8. PERFORMING ORGANIZATION REPORT NO.

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Air Force Research Laboratory (AFMC)
AFRL/RQR
5 Pollux Drive
Edwards AFB CA 93524-7048

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S REPORT NUMBER(S)
AFRL-RZ-ED-VG-2012-221

12. DISTRIBUTION / AVAILABILITY STATEMENT
Distribution A: Approved for Public Release; Distribution Unlimited. PA#12471

13. SUPPLEMENTARY NOTES
Conference paper for the 2012 AIAA Fluids conference, New Orleans, Louisiana in 25 June 2012.

14. ABSTRACT

Behavior of unconfined transverse jets has been studied extensively, but little work is reported on the flow characteristics of confined transverse jets. The latter has been numerically investigated using a number of RANS codes. The computational results obtained from these codes have been evaluated against the existing experimental data, and the results of a Large-Eddy Simulations (LES) code reported in the literature. Furthermore, an extensive validation effort has been conducted to characterize the performance of the codes for predicting the flow within a propulsion-related mixing configuration. The validation case involves eight circumferentially spaced transverse jets issuing radially into an axisymmetric main flow, a configuration relevant for gas turbine burners and new liquid rocket engine preburners. The main flow Reynolds number was 1.7 x 105 and the jet-to-main flow momentum flux ratio was sixteen. The momentum and scalar mixing was investigated through the solution of the Reynolds-Averaged Navier Stokes (RANS) equations. The solutions of three commercial RANS solvers, Fluent, STAR-CCM+, and CFD++, are compared to experimental data and large-eddy simulation (LES) results available in literature. Due to demonstrated periodicity, only a one-eighth pie-shaped section of the geometry was considered. The different commercial codes used the same geometry, grid, boundary conditions, and variations of the k-_ turbulence model. The LES results obtained from literature used a different grid, but the same geometry. All numerical simulations using the above mentioned codes capture salient flow structures such as the counter-rotating vortex pair (CRVP). Experimental data used for validation of the codes\ include mean axial velocity and jet fluid mixture fraction profiles (at three distinct axial locations), jet trajectory, turbulent kinetic energy distributions, and velocity and mixture fraction cross-plane distributions. All CFD results except CFD++, exhibit symmetrical solutions about the center plane. The current investigation shows that although all codes considered predict the experimental data with various degrees of accuracy, Fluent using the standard k-_ turbulence model with the standard ll function, and LES results compare exceptionally well with the experimental data for this flow regime and configuration.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**	SAR	22	Nils Sedano
Unclassified	**Unclassified**	**Unclassified**			**19b. TELEPHONE NO** *(include area code)* 661-275-5972

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. 239.18

THE AIR FORCE RESEARCH LABORATORY
LEAD | DISCOVER | DEVELOP | DELIVER

The World's Forum for Aerospace Leadership

NUMERICAL SIMULATION OF CONFINED MULTIPLE TRANSVERSE JETS

25 June 2012
Presented to: AIAA Fluids Meeting, New Orleans, Louisiana

Farhad Davoudzadeh, Ph.D. – Air Force Research Laboratory
David Forliti, Ph.D. - Jackson and Tull, LLC
Anh-Tuan Le, Ph.D. & Henry Vu, Ph.D. - Advatech Pacific, Inc.

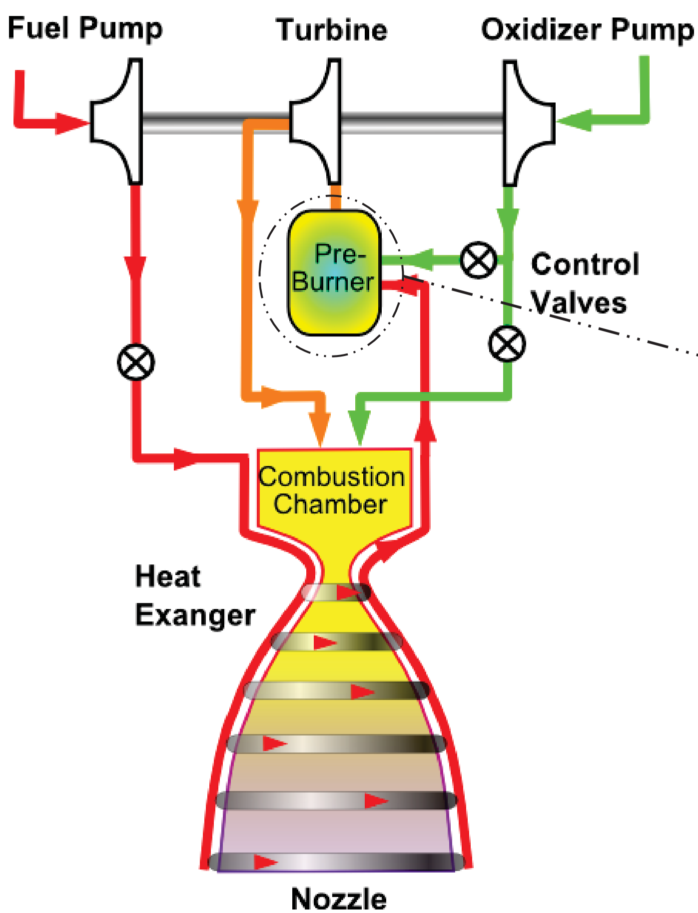
Outline:

- Objectives
- Jet in Crossflow characteristics
- CFD Validation Case and the experimental data
- Geometry, Grid, and the Boundary Conditions
- Comparative Analysis Results
- Summary

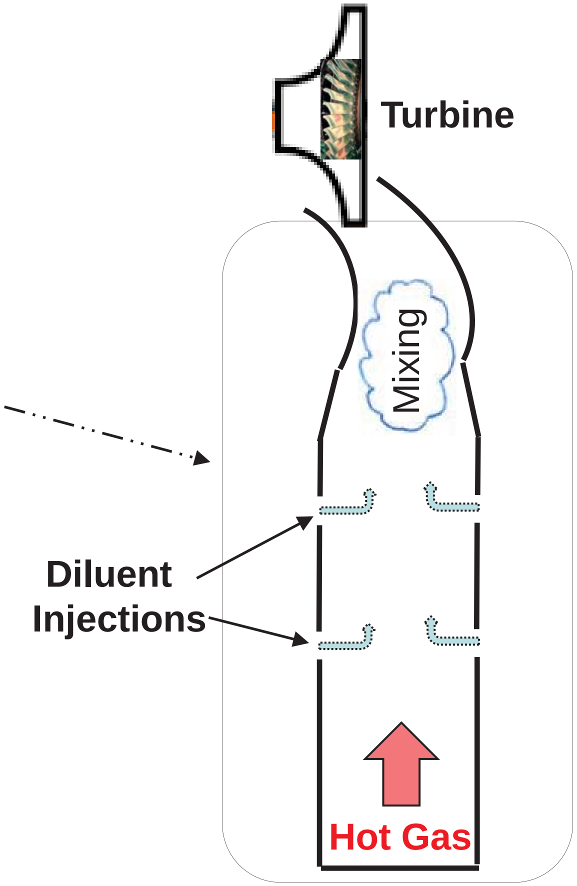
Objectives:

- **Primary Objectives:**
  - **Investigate mixing characteristics of propellants in preburners**
  - **CFD support for the in-house Themis project**
- **Secondary Objective:**
  - **Validate commercial CFD codes—Fluent, CFD++, and Star-ccm++ against experimental data and an LES results**
  - **Provide numerical data for theoretical developments**

Staged Combustion Cycle Rocket Engine:



Staged Combustion cycle



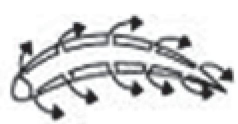
Pre-burner Injector

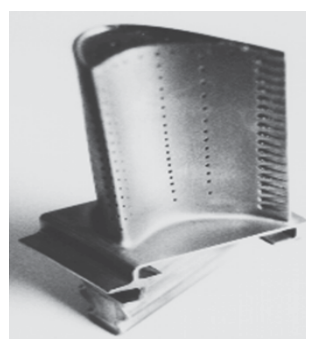
Temperature Distribution On the Turbine Blades

Jet in Cross Flow Characteristics

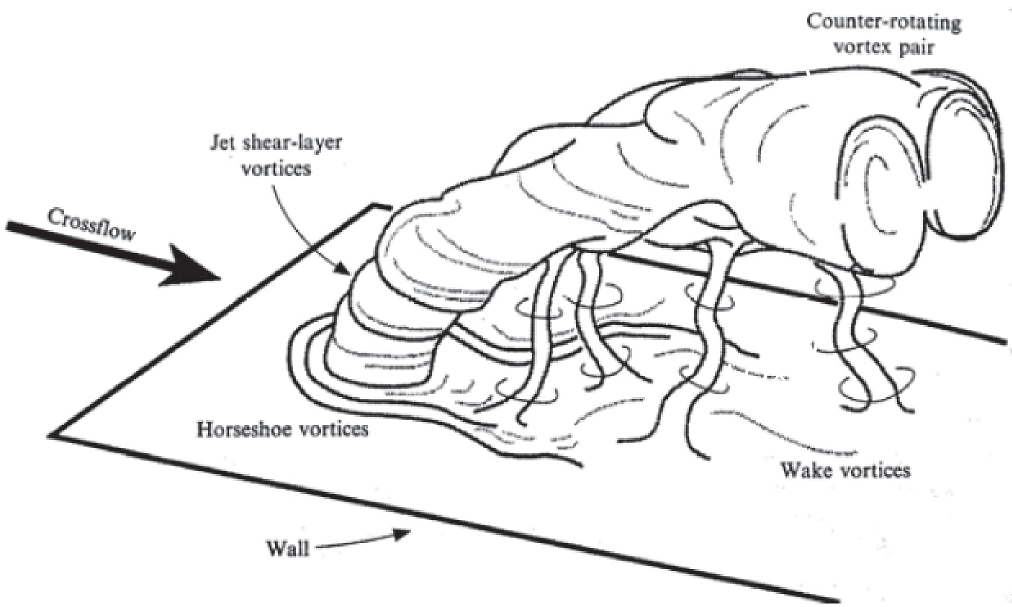
Complex flow with multiple engineering applications:

- ❑ Liquid rocket engine pre-burner
- ❑ Gas turbine combustors
- ❑ Film cooling of turbine blades
- ❑ Air pollution
- ❑ Chimney emissions
- ❑ Industrial burners
- ❑ Chemical mixing
- ❑ Wastewater discharges





Jet in Cross Flow Characteristics



Adapted from Fric and Roshko (1994)

Project Themis:

- In-house combustion device research within the Liquid Rocket Engines Branch (LREs) of AFRL

- Focuses on investigation of LOX/Hydrocarbon high pressure combustion devices through:
  - Theory development
  - Modeling and simulation
  - Subscale experimentation at reacting and inert conditions:
    - Design & test a 10k lbf pre-burner for liquid propellants
      - Requires good mixing of variable density propellants
      - Temperature uniformity
      - Concentration uniformity

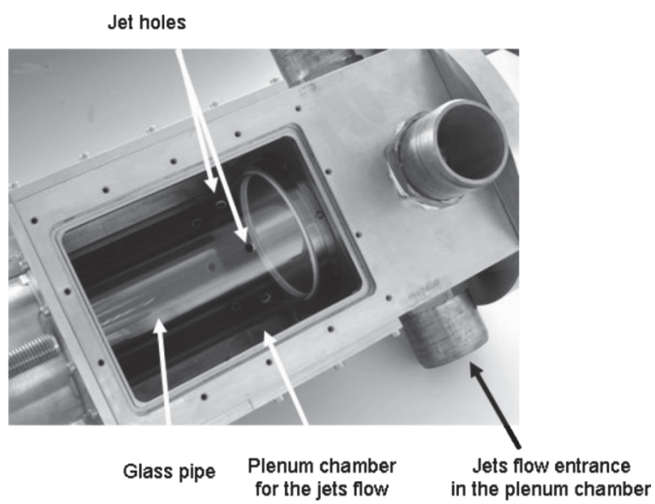
Jet in Crossflow

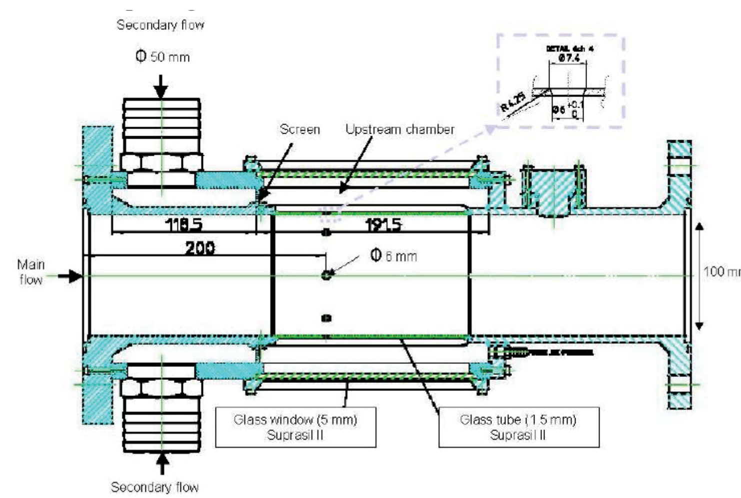
CFD Validation Case

Conduct a validation study for a baseline case:

- Experimental data available
- Unity density ratio
- Multiple confined transverse jets
- Single phase/component

ONERA experimental/LES studies of an eight jet mixing chamber

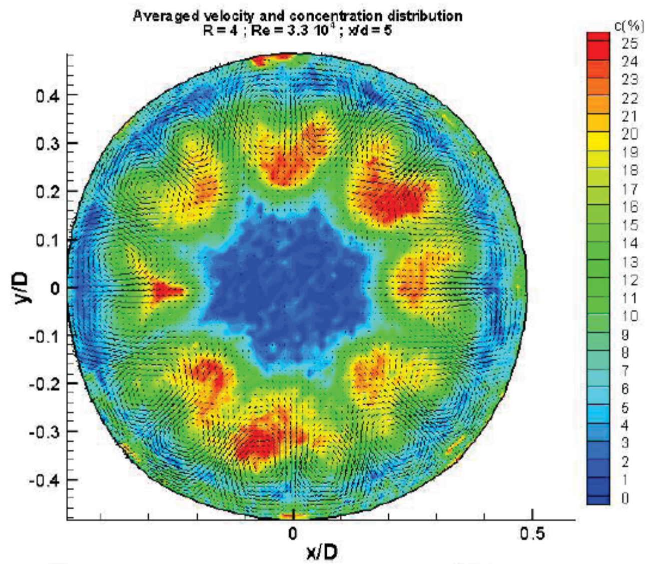


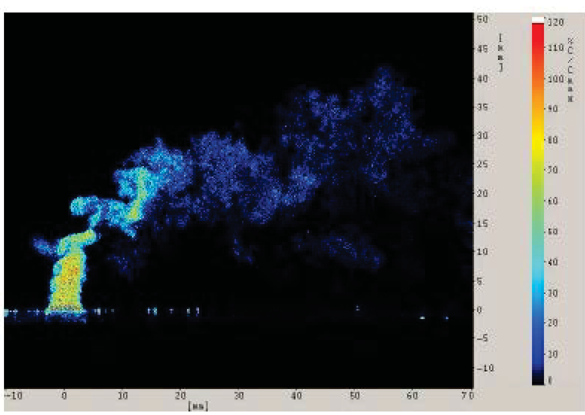


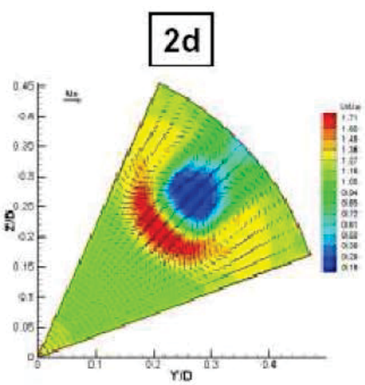
ONERA Experiments

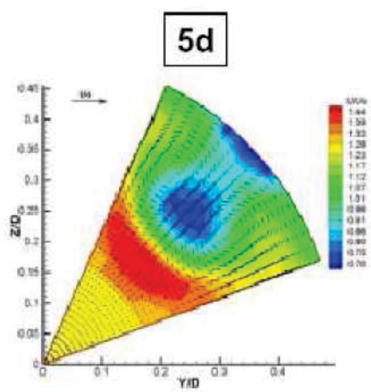
Pros:
- PIV and PLIF data
- Characterized boundary conditions

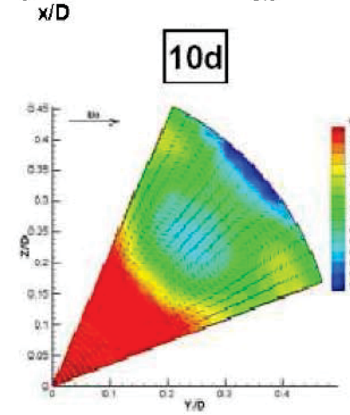
Cons:
- ONERA unable to share unpublished data
- Limited flow conditions (two cases)
- Emphasis on near field (> 1 main pipe diameter)
- Geometry not fully specified



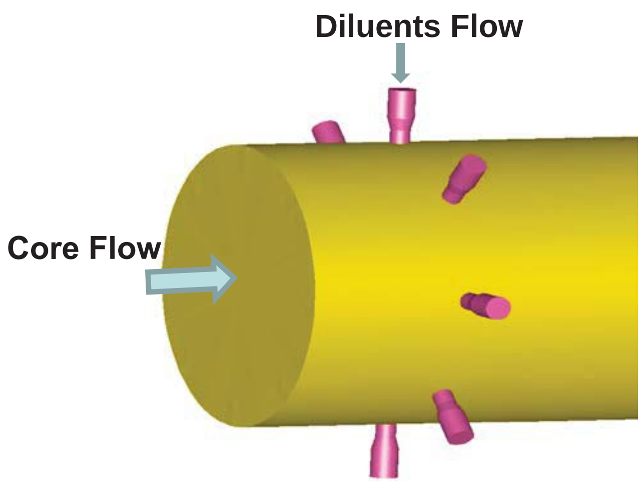


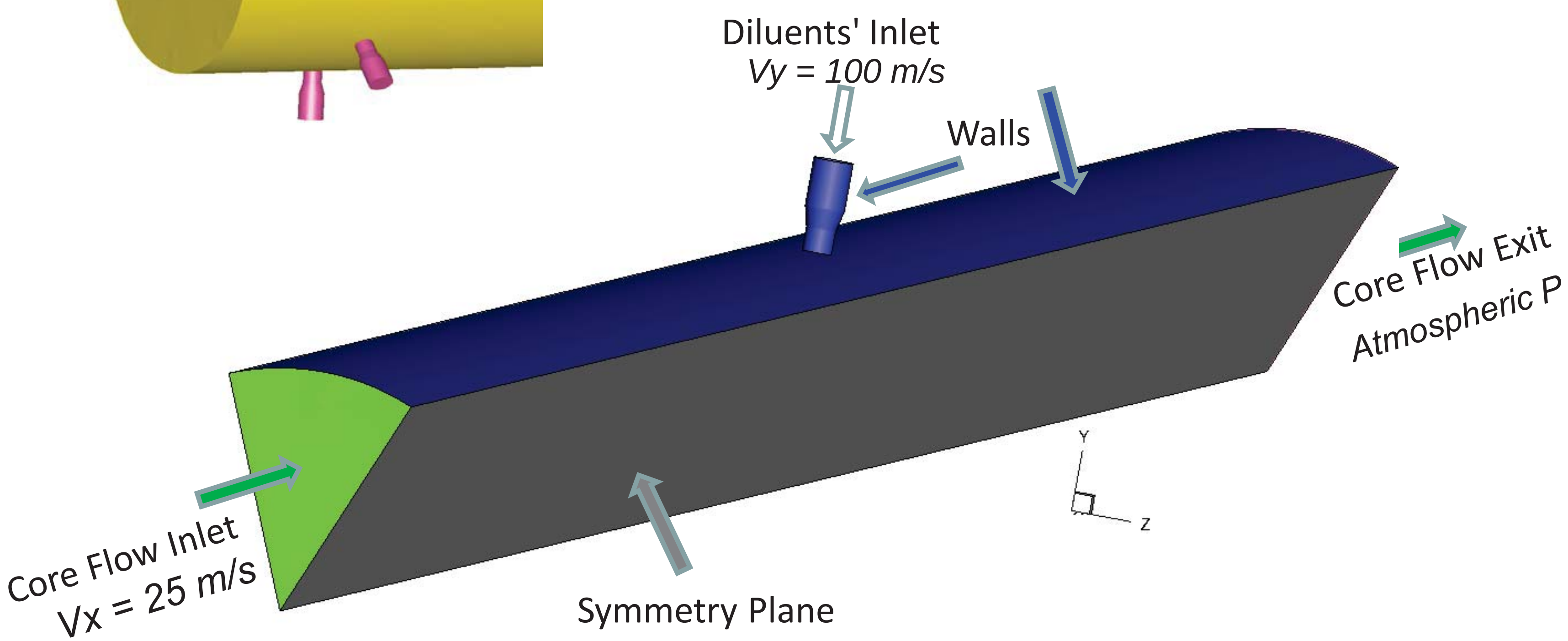




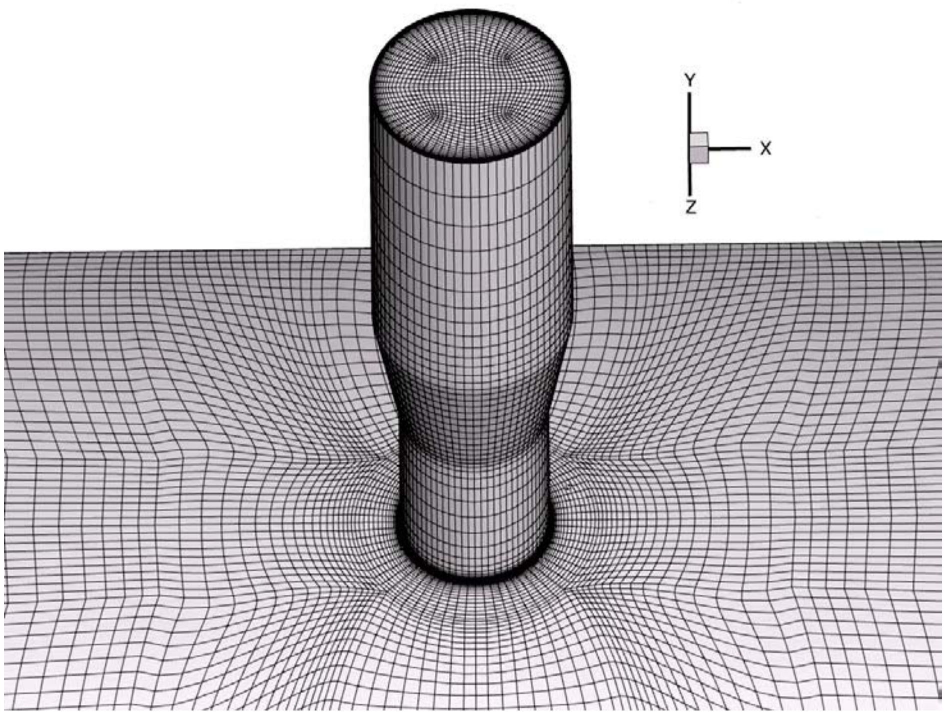


Geometry Model

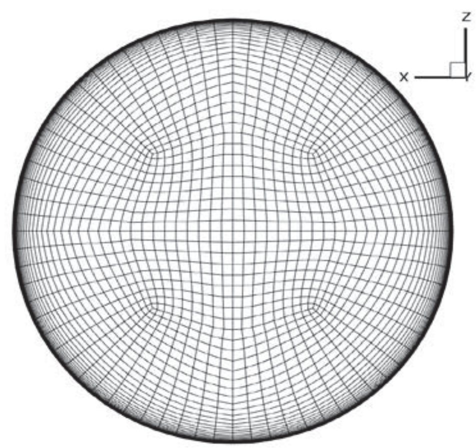




Grid: 800K elements, all hex

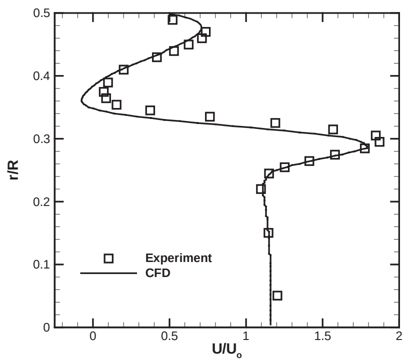


Jet Inlet

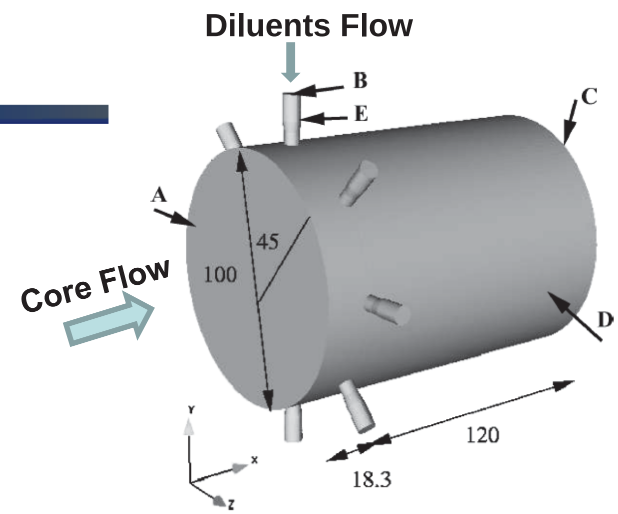


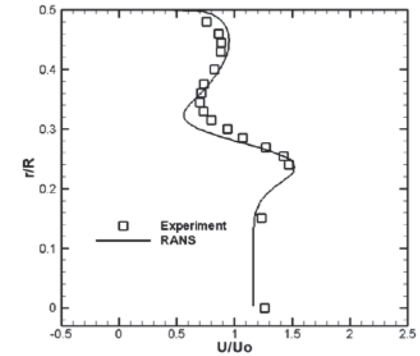
Inlet and exit

Numerical and Experimental Investigation of Jets in Cross Flow

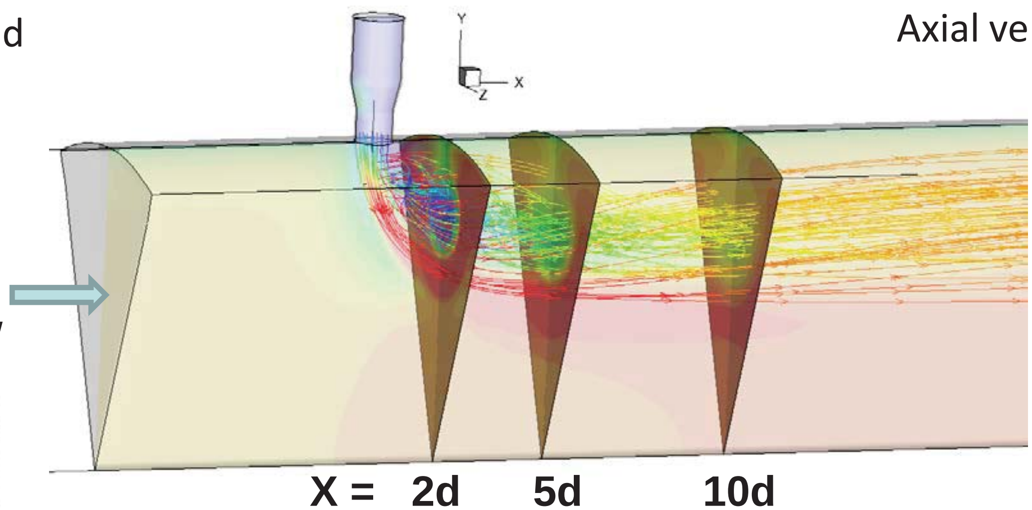


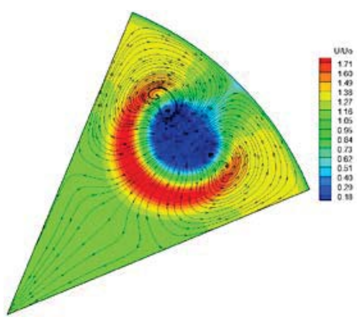
Axial velocity profile, x = 2d



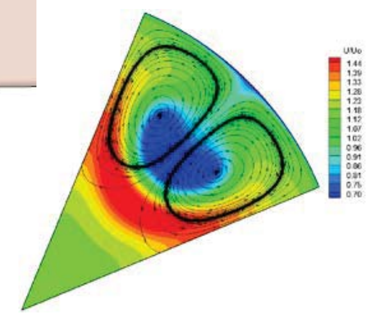


Axial velocity profile, x = 5d



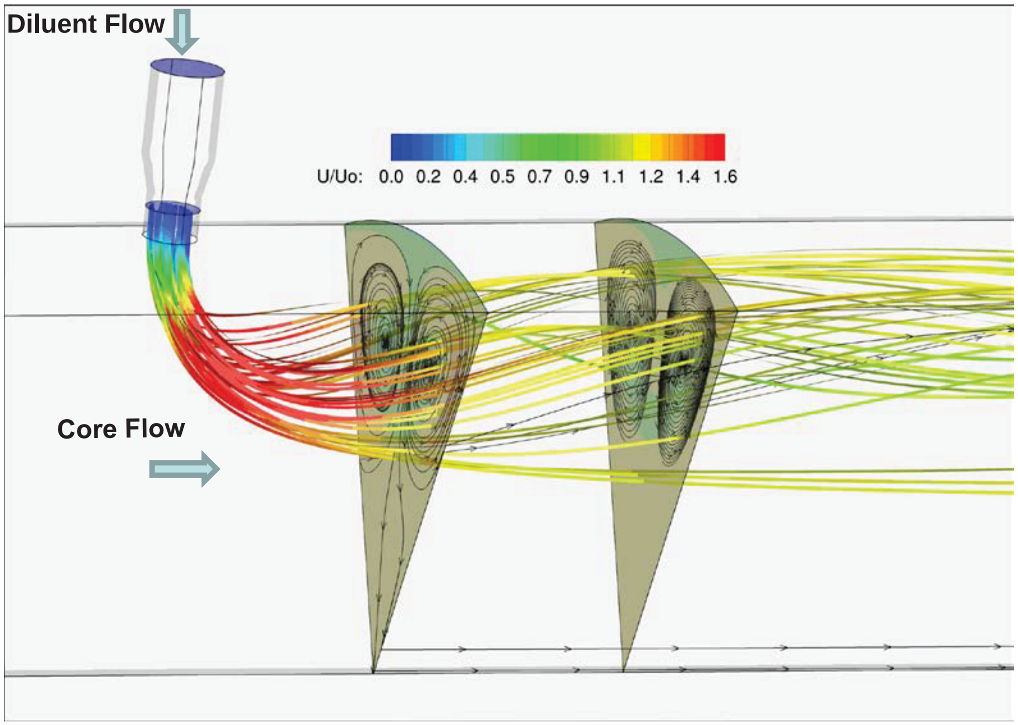


Axial velocity Contours, x = 2d

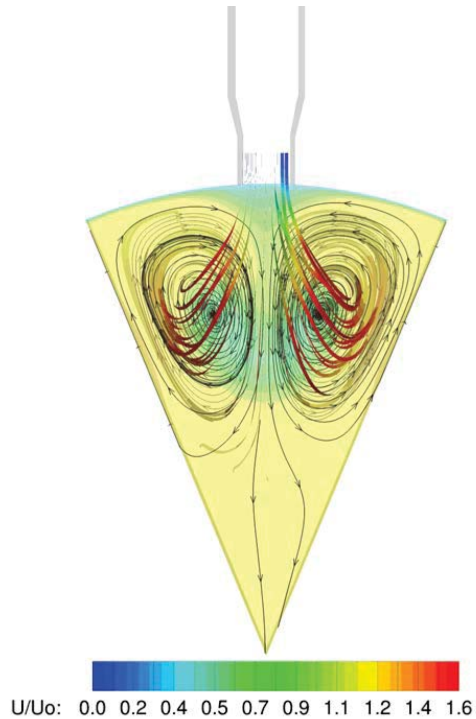


Axial velocity contours, x = 5d

Counter Rotating Vortex Pairs



Perspective view



View from Upstream

Experimental data versus computational results: Axial Velocity Contours

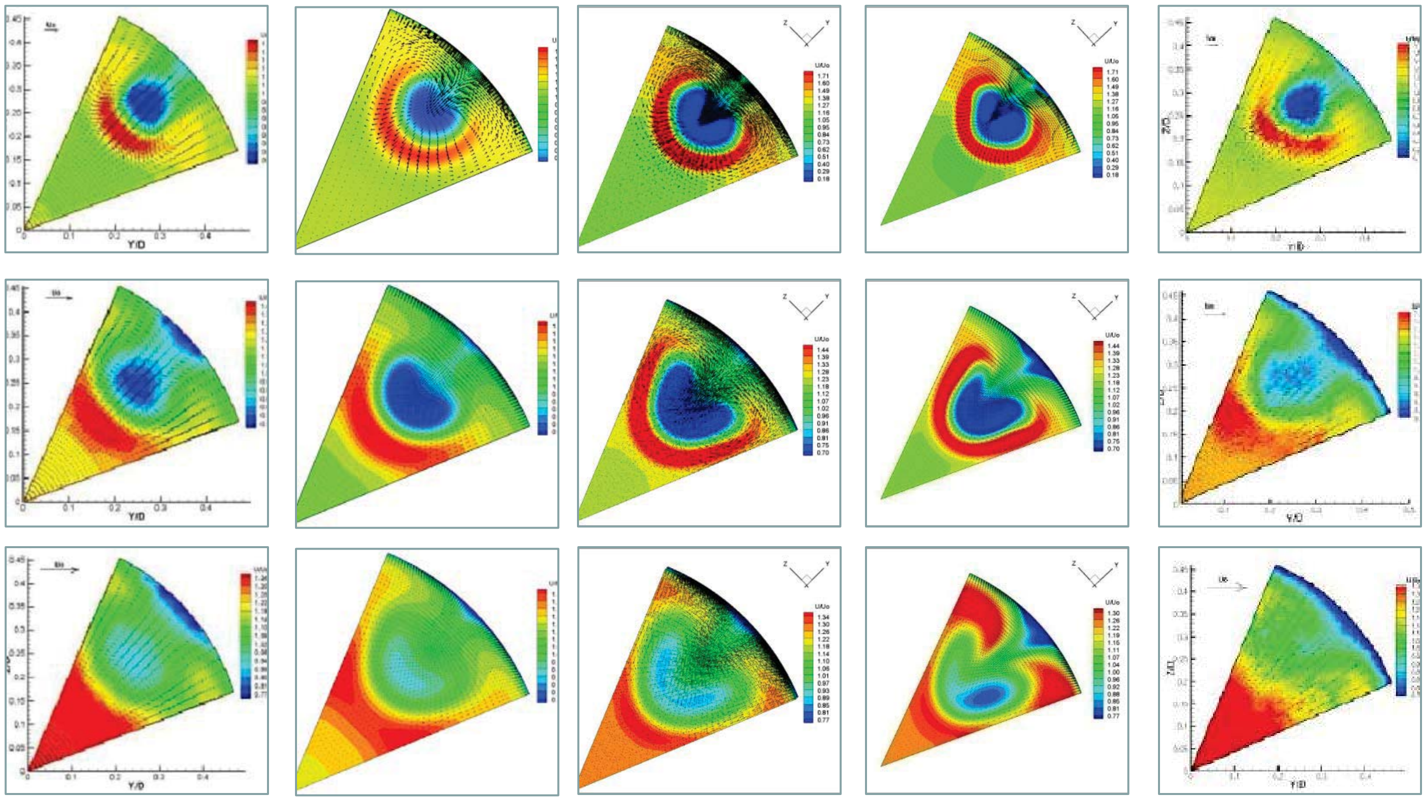
Axial locations (d = inj. dia.)

Experiment | Fluent | STAR-CCM+ | CFD^{++} | LES (ONERA)

X = 2d:

X = 5d:

X = 10d:



Experimental data versus computational results: Axial Velocity Profiles

Axial locations (d = inj. dia.)

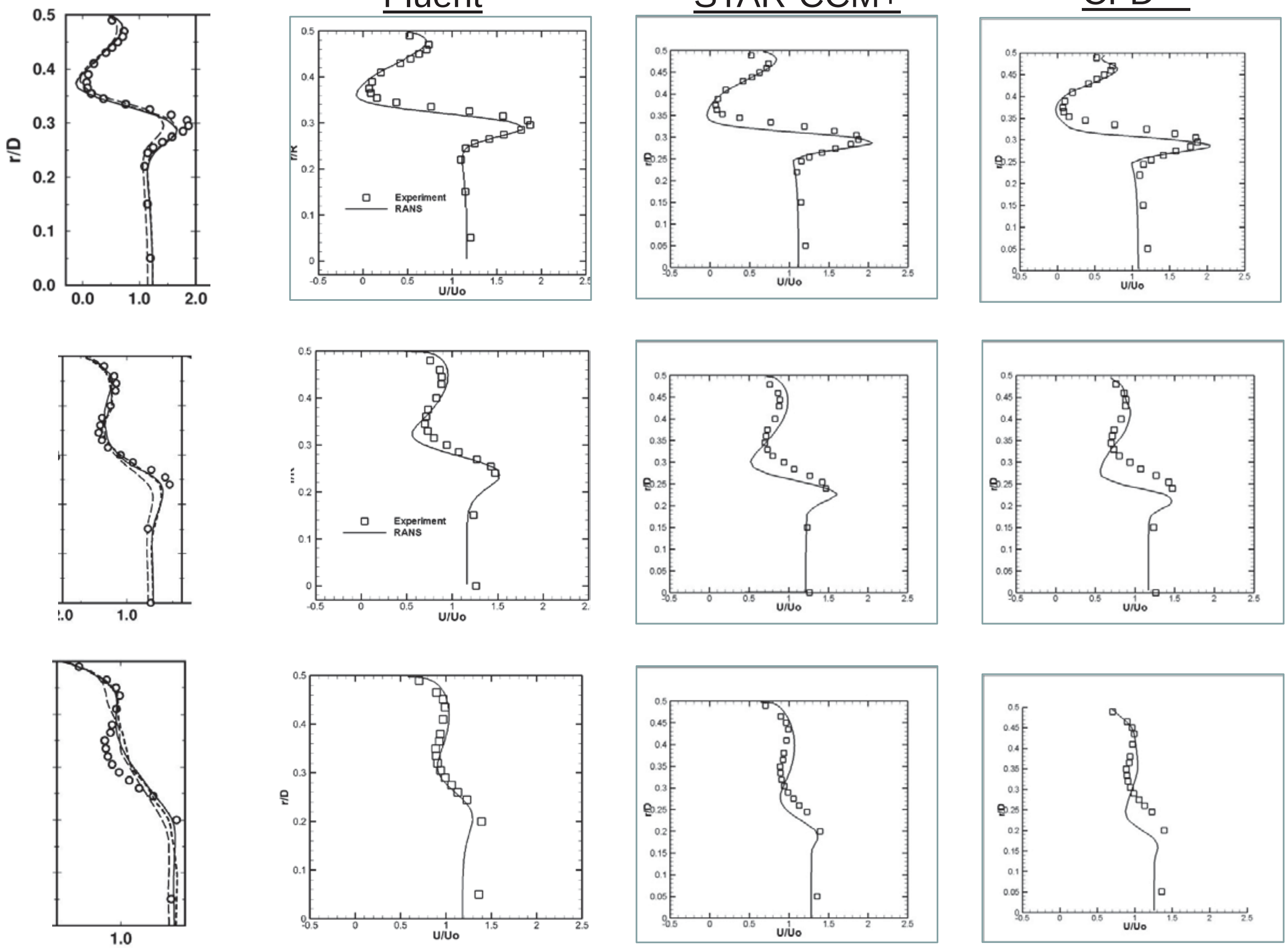
⇩

LES (ONERA) | Fluent | STAR-CCM+ | CFD^{++}

X = 2d:

X = 5d:

X = 10d:



Distribution A: Approved for public release; distribution unlimited

Experimental data versus computational results: Axial Velocity Profiles

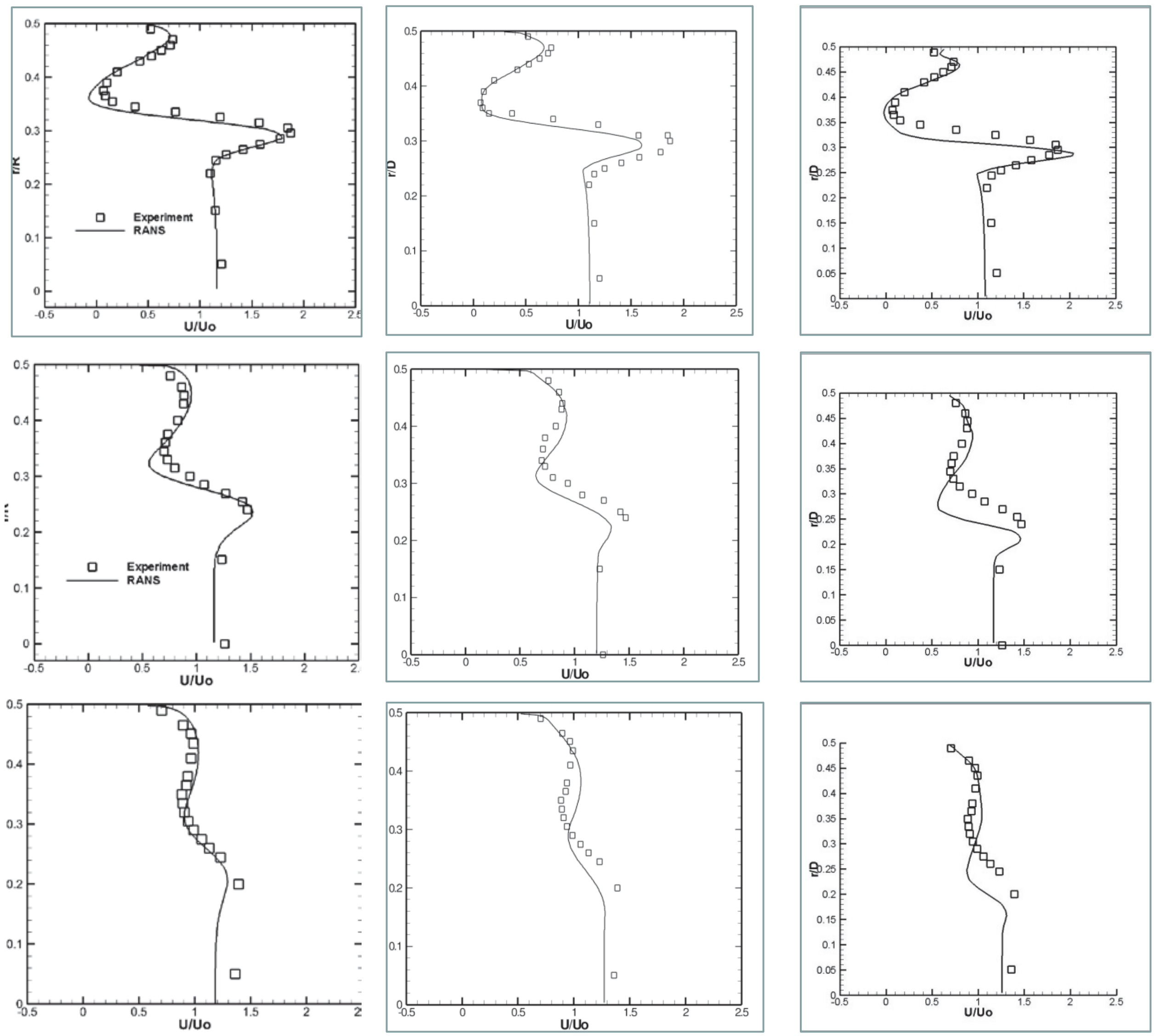
Axial locations (d = inj. dia.)

Fluent: Std k-ϵ | Fluent Realizable k-ϵ | CFD++ Realizable k-ϵ

X = 2d:

X = 5d:

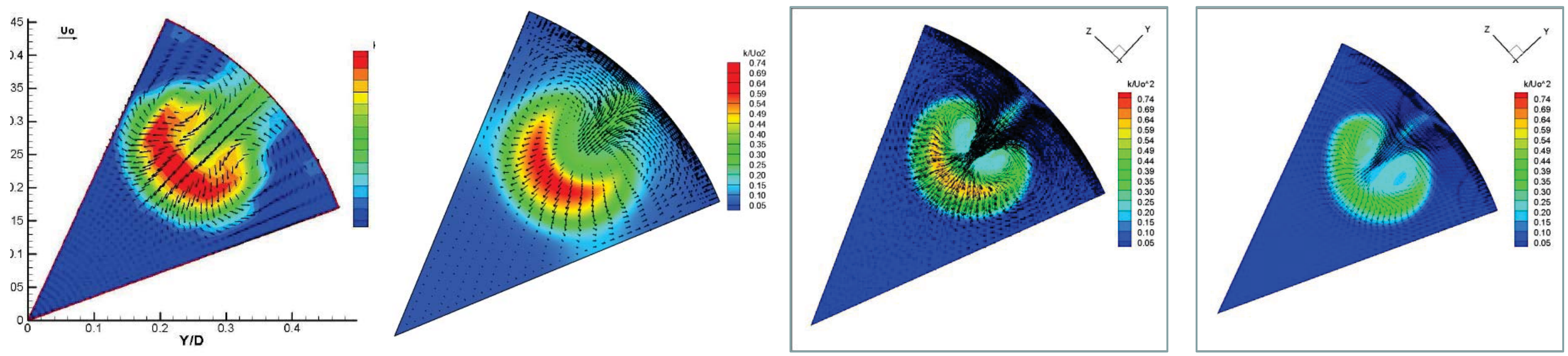
X = 10d:



Experimental data versus computational results:

	Experiment	Fluent	STAR-CCM+	CFD^{++}

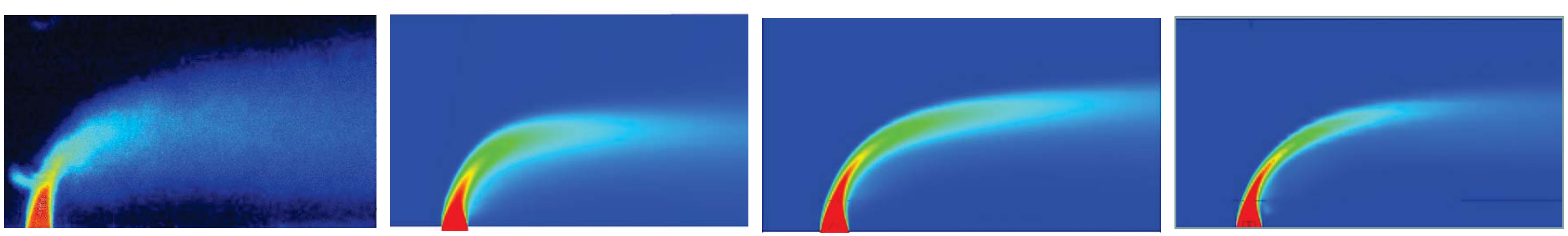
X = 2d:



Turbulent kinetic energy contours

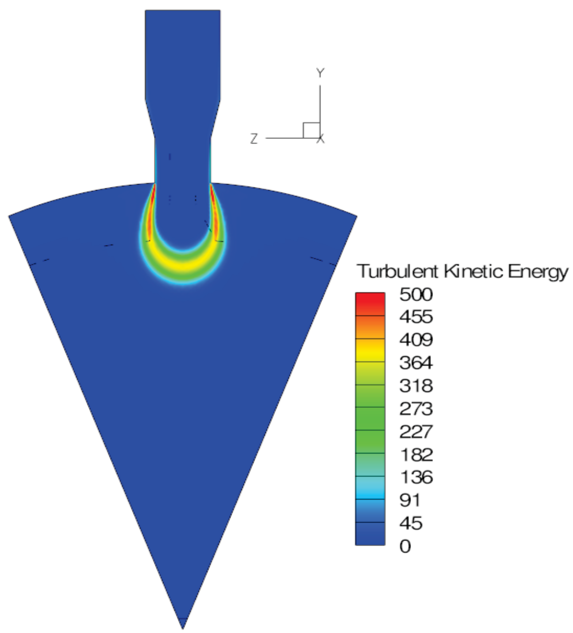
	Experiment	Fluent	STAR-CCM+	CFD^{++}

Z = 0:

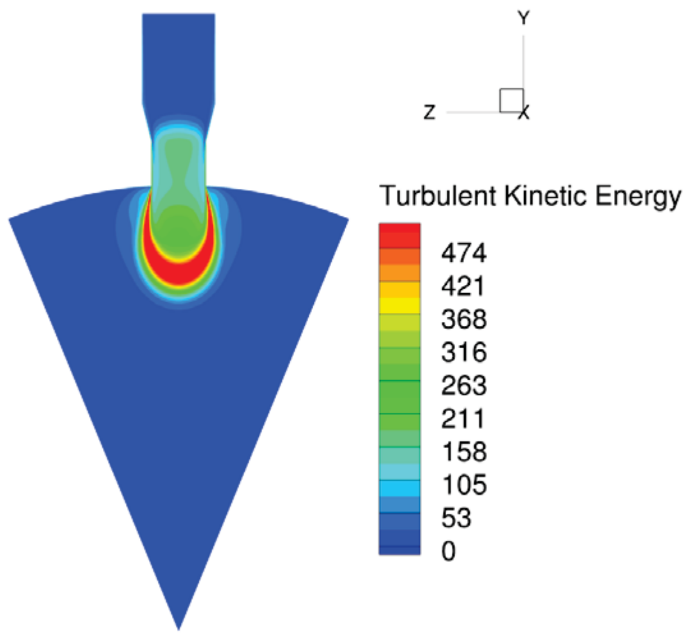


Jet Species Mass Fraction

Turbulent Kinetic Energy Produced via Two Turbulence Models



(a)



(b)

Realizable k-ϵ

Standard k-ϵ

Experimental data versus computational results: Jet Concentration Profiles

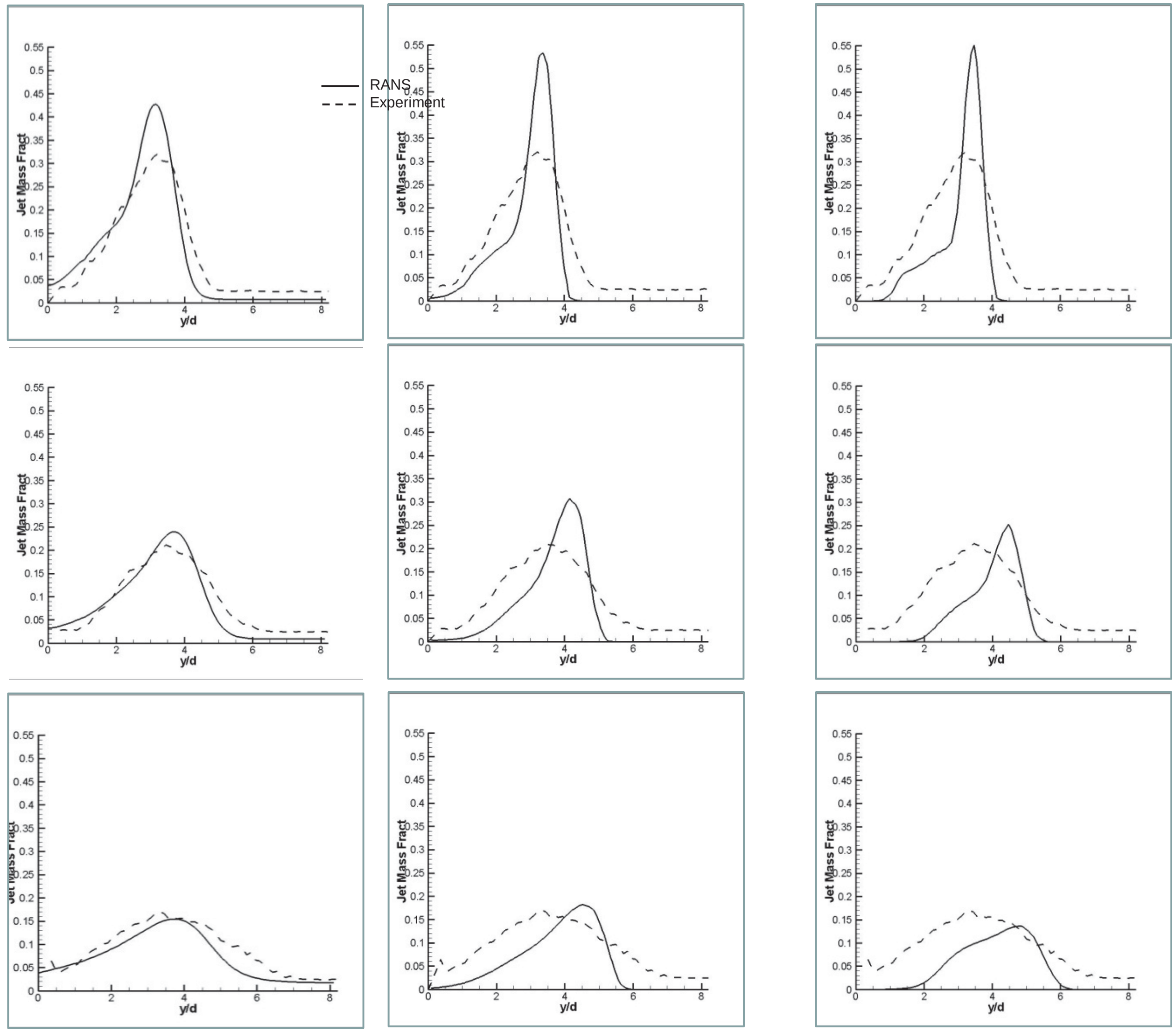
Axial locations (d = inj. dia.)

Fluent | STAR-CCM+ | CFD++

X = 2d:

X = 5d:

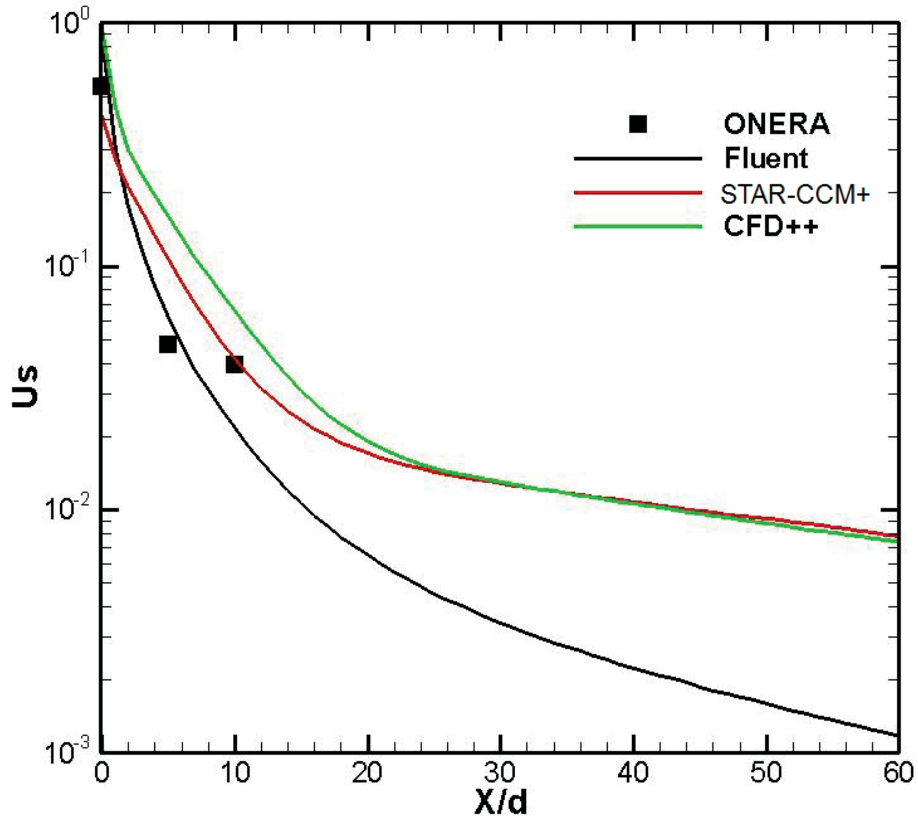
X = 10d:



Comparison of Predicted Un-mixedness

Prediced Un-mixedness over cross-sectional area for three codes

$$U_s = \frac{\sigma_C^2}{\overline{C}\left(1-\overline{C}\right)} = \frac{1/A\int\left(C-\overline{C}\right)^2 dA}{\overline{C}\left(1-\overline{C}\right)}$$



Summary

- All codes (Fluent, CFD++, and Star-ccm+ using realizable k-ϵ) predict the experimental centerline velocity profiles reasonably well
- Fluent, using the standard k-ϵ, & LES predict turbulent kinetic energy, centerline velocity profiles and centerline concentration profiles that better match the experimental data
- CFD++ calculations predict jet flow that manifests itself in an asymmetric wake profile in a symmetric domain
- STAR-CCM+ and FLUENT calculations predict a more stable flow that maintains wake symmetry
- Fluent predicts higher overall mixedness than other codes